COMPASS
www.compassis.com

Tdyn-CFD+HT - Validation Case 1

Two dimensional cavity flow

Tdyn

Version
15.1.0

http://www.compassis.com
info@compassis.com
November 2018

Table of Contents

1 Validation Case 1 - 2D cavity flow

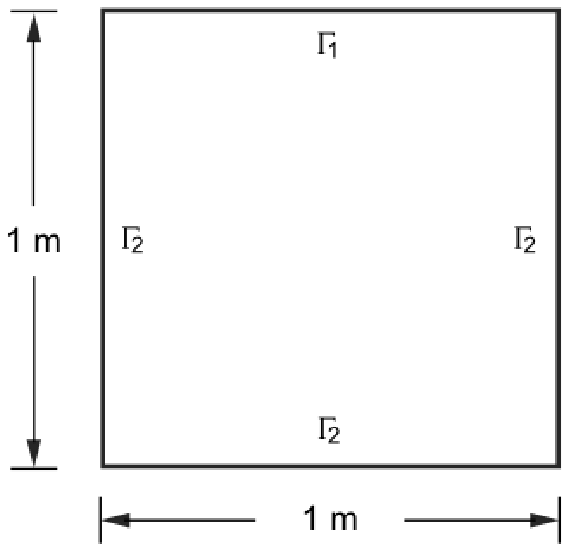
The driven cavity flow is a very well known benchmark problem, which has been studied extensively in the literature (i.e. reference [1]).

The steady incompressible cavity flow problem (Navier-Stokes equations) is performed in a 2D squared domain Ω of 1 meter wide by 1 meter high. Despite its simple geometry, the driven cavity flow retains a rich fluid flow physics manifested by multiple rotating recirculating regions on the corners of the cavity, depending on the Reynolds number.

This validation case concerns a fluid contained in the analysis domain, with a Dirichlet boundary conditions for velocity on the upper boundary, which is moving with velocity tangent to the boundary. The rest of the boundaries of the cavity remain stationary. Numerical simulations are computed for Re<21,000 and results are compared against the benchmark solutions given in reference [1].

It is important to note that fluid wall is used to define fluid boundary properties in an automatic way. A brief summary of the boundary conditions that have been applied to the analysis domain is given as follows:

Condition	Boundary
Fix velocity	Γ_1
Fluid wall	Γ_2



Fix velocity and fluid wall boundary conditions

Problem description

The problem consists of the analysis of a two-dimensional cavity flow, with the following characteristics:

- User defined problem
  Simulation dimension: 2D plane
  Multi-physics analysis: Fluid flow

- Geometry
  1.0x1.0 meters squared domain.

- Domain
  Steady-state, stationary.

- Material properties
  Density ρ=1 kg/m^3
  Viscosity μ=0.001 kg/(m·s)

- Fluid properties
  Fluid properties are adjusted to match the required Reynolds numbers. Since the fluid is assumed incompressible in all cases under analysis, Re is the only nondimensional number governing the fluid flow behaviour. Density ρ is modified for each case, while the shearing velocity and the size of the cavity are kept constant.

Re	ρ [Kg/m^3]	v [m/s]	L [m]	μ [kg/m·s]
1000	1.0	1.0	1.0	0.001
2500	2.5	1.0	1.0	0.001
5000	5.0	1.0	1.0	0.001
7500	7.5	1.0	1.0	0.001
10000	10.0	1.0	1.0	0.001

- Fluid Models
  Incompressible.

- Boundary Conditions
  Fix velocity: the velocity vector is fixed at the upper boundary of the domain. The velocity component tangent to the boundary is fixed to 1.0 m/s, whereas the velocity component

normal to the boundary is fixed to 0.0 m/s.

Such a boundary condition resembles a fluid flow passing through the surface of the cavity thus shearing the fluid inside.

Wall/Body: VFixWall condition has been used in order to enforce the no-slip condition on the vertical boundaries and on the lower boundary.

- Initial conditions
  Velocity: initialized to the value 0.0 m/s for the whole model domain.

  Pressure: automatically initialized to 0.0 Pa.

  Turbulence model: all cases under analysis were run assuming a laminar flow regime.

- Solver parameters
  The fractional step solver is chosen for all the simulations.

  Assembling type: mixed.

  Time step: 0.1 s

  Non-symmetric solver: Bi-conjugate Gradient (tolerance = 1.0E-07) with ILU preconditioner.

  Symmetric solver: Conjugate Gradient (tolerance = 1.0E-07) with ILU preconditioner.

Mesh

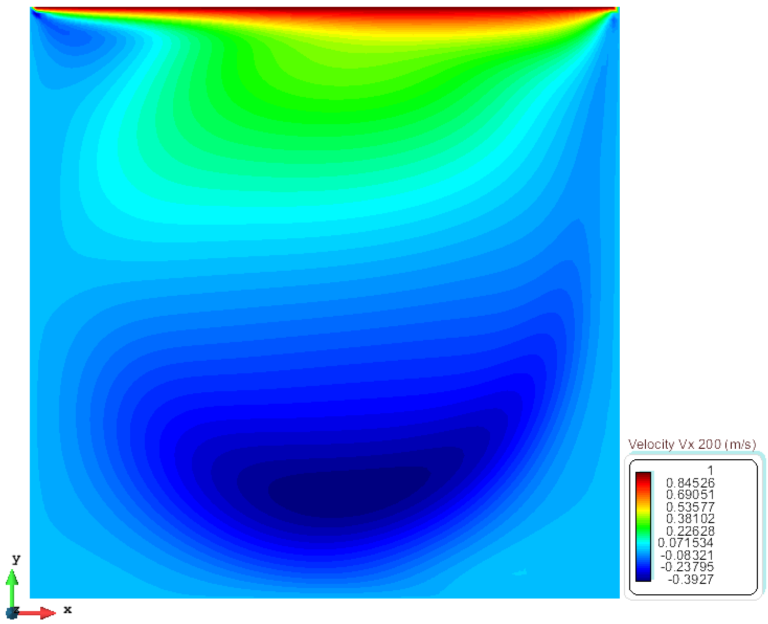
The two-dimensional space domain Ω is discretized by a 100x100 structured grid of linear quadrilateral elements. The resulting finite elements mesh has 10201 nodes, and 10400 linear quadrilateral elements.

Results

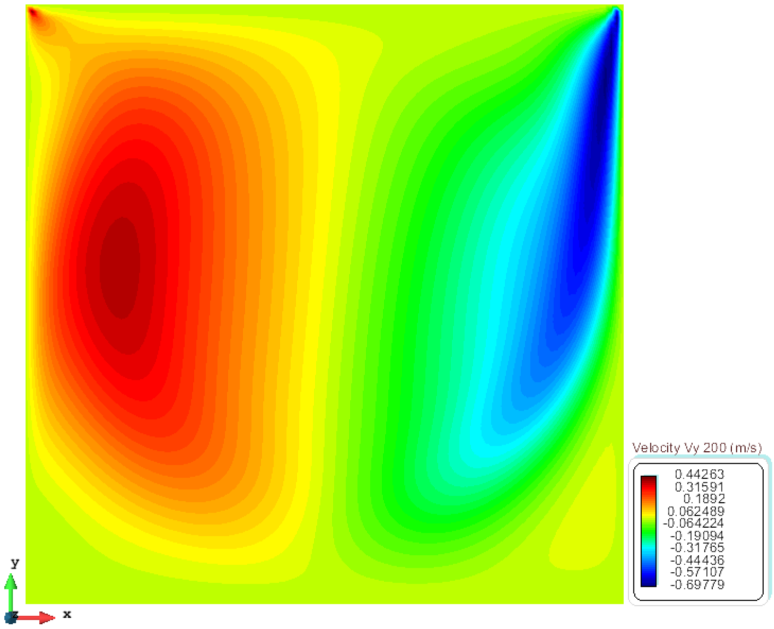
The results given below correspond to the velocity and pressure distributions on the domain Ω, once the solution has reached the steady state (t=0.1s) for Re=1000.

In addition, it has been checked that the steady state is reached at the same time step (t=0.1s) for various Reynolds numbers (Re = 1000, 2500, 5000, 7500, 10000).

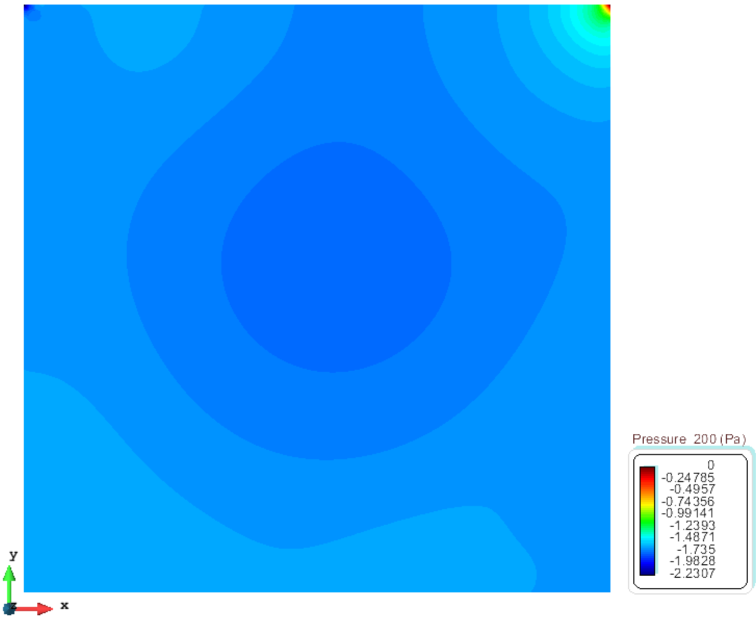
The solution of the benchmark case given in ref. [1] has been used in this case to validate the solution obtained with Tdyn.



x-velocity distribution at the last time step (t=0.1s), Re=1000



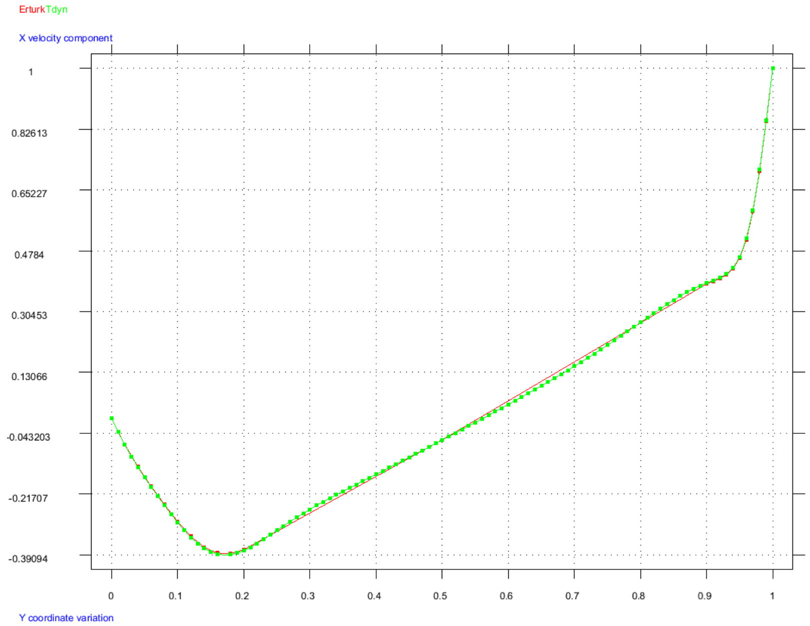
y-velocity distribution at the last time step (t=0.1s), Re=1000



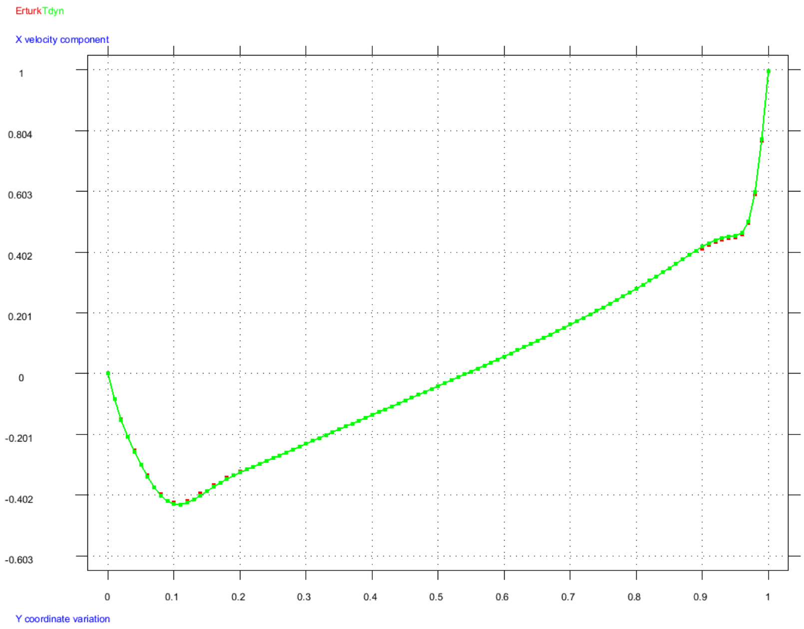
Pressure distribution at the last time step (t=0.1s)., Re=1000

Simulations were performed for various Reynolds numbers (Re = 1000, 2500, 5000, 7500, 10000). For each case, the computed u-velocity profile along a vertical line passing through the geometric centre of the cavity is compared against other simulations performed in reference [1].

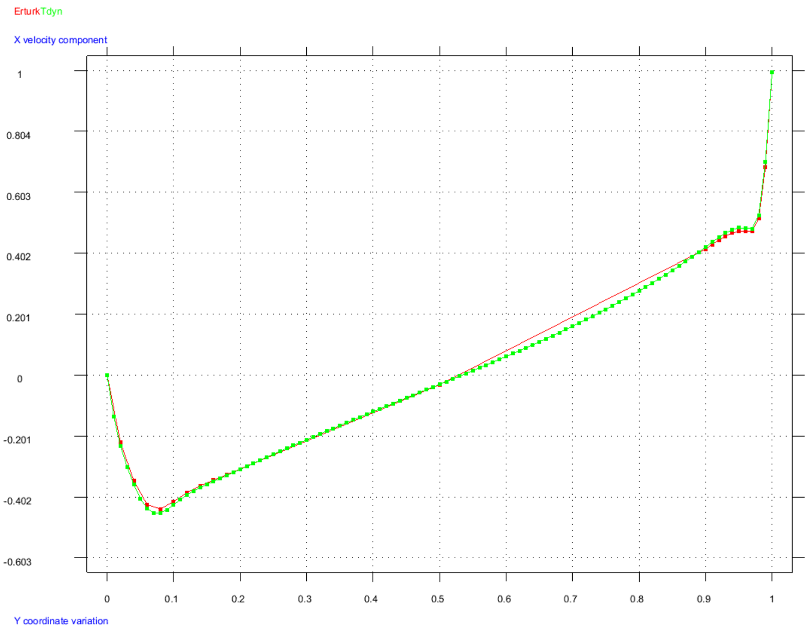
The figures below show the difference between the benchmark given in reference [1] and Tdyn computation, for each Reynolds number.



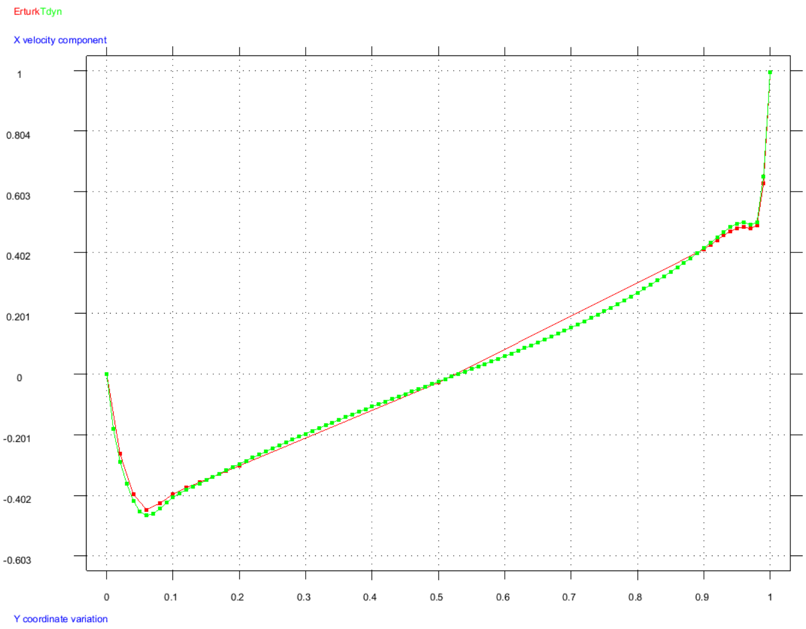
X-velocity vs. Y coordinate variation, Re=1000



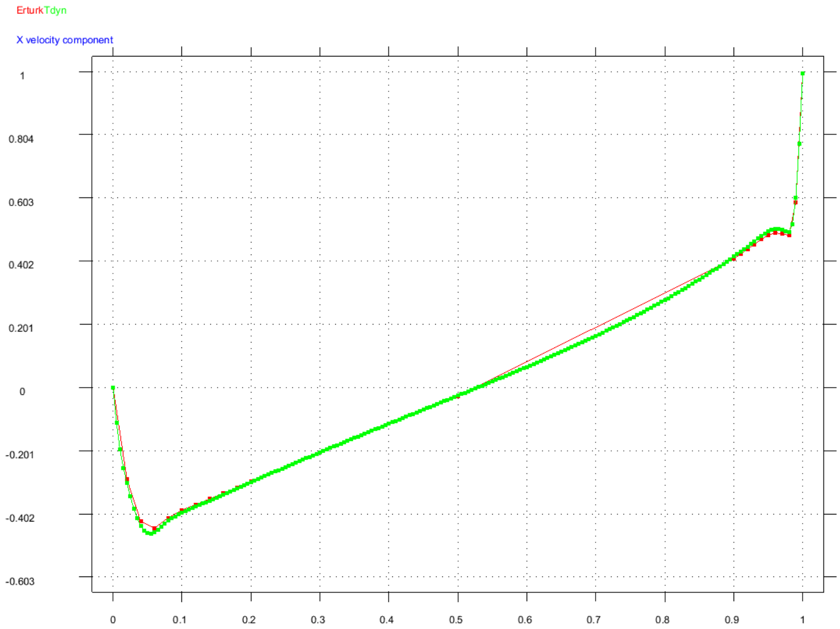
X-velocity vs. Y coordinate variation, Re=2500



X-velocity vs. Y coordinate variation, Re=5000



X-velocity vs. Y coordinate variation, Re=7500



X-velocity vs. Y coordinate variation, Re=10000

It must be emphasized that Tdyn offers a very good agreement to the benchmark results given in reference [1], for a wide range of Reynolds numbers (Re = 1000, 2500, 5000, 7500, 10000).

References

[1] E.Erturk, T.C.Corke and C.Gökçöl. Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers. Int. J. Numer. Meth. Fluids 2005; 48:747-77

Validation Summary

CompassFEM version	15.1.0
Tdyn solver version	15.1.0
RamSeries solver version	15.1.0
Benchmark status	Successfull
Last validation date	27/11/2018